

Appl. No. 09/438,856
Amdt. Dated 08/18/2004
Reply to Office Action of 03/19/2004

BEST AVAILABLE COPY

Appendix II

SMPS Simulation with SPICE3

by Steven M. Sandler

Copyright 1997

pages 10-15

SMPS Simulation with SPICE 3

Steven M. Sandler

Library of Congress Cataloging-in-Publication Data

Sandler, Steven M.
SMPS simulation with SPICE 3 / Steven M. Sandler.
p. cm.

Includes bibliographical references and index.
ISBN 0-07-057853-2 (hardcover)

1. Switching power supplies—Computer simulation. 2. SPICE
(Computer file). I. Title.
TK7881.15.S26 1997
621.381'044—dc21

96-37203
CIP

McGraw-Hill



A Division of The McGraw-Hill Companies

Copyright © 1997 by The McGraw-Hill Companies, Inc. All rights reserved. Printed in the United States of America. Except as permitted under the United States Copyright Act of 1976, no part of this publication may be reproduced or distributed in any form or by any means, or stored in a data base or retrieval system, without the prior written permission of the publisher.

2 3 4 5 6 7 8 9 0 DOC/DOC 9 0 1 0 9 8 7

P/N 057853-2

PART OF

ISBN 0-07-913227-8

The sponsoring editor for this book was Steve Chapman, the editing supervisor was Marc Campbell, and the production supervisor was Don Schmidt.

Printed and bound by R. R. Donnelley & Sons Company.

McGraw-Hill books are available at special quantity discounts to use as premiums and sales promotions, or for use in corporate training programs. For more information, please write to the Director of Special Sales, McGraw-Hill, 11 West 19th Street, New York, NY 10011. Or contact your local bookstore.



This book is printed on recycled and acid-free paper containing a minimum of 50% recycled deinked fiber.

Information contained in this work has been obtained by The McGraw-Hill Companies, Inc. ("McGraw-Hill") from sources believed to be reliable. However, neither McGraw-Hill nor its authors guarantee the accuracy or completeness of any information published herein and neither McGraw-Hill nor its authors shall be responsible for any errors, omissions, or damages arising out of use of this information. This work is published with the understanding that McGraw-Hill and its authors are supplying information but are not attempting to render engineering or other professional services. If such services are required, the assistance of an appropriate professional should be sought.

McGraw-Hill

New York San Francisco Washington, D.C. Auckland Bogotá
Caracas Lisbon London Madrid Mexico City Milan
Montreal New Delhi San Juan Singapore
Sydney Tokyo Toronto

24		0	time	2.00
----	--	---	------	------

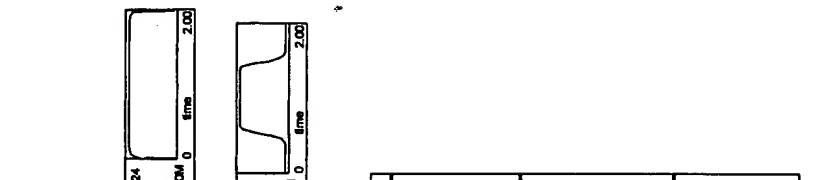
0	time	200
---	------	-----

[illegible]

hysteresis (selem), ion switch (PSW1)

- de voltages and*

A DC analysis, locally performed transient initial Bias Solution",

hysteresis (selem),
tion switch (PSW1).

de voltages and

ing point of the A DC analysis, locally performed transient initial Bias Solution", determine the lin-

earized, small-signal models for all non-linear devices. It should be noted that these two operating point calculations can be different, depending on the DC and transient stimulus which is used.

Transfer Function Analysis

Produces a small signal DC transfer function.

The transfer function analysis calculates the small signal ratio of the output node to the input source, and also the input and output impedances of a circuit. This analysis may be used to determine the small signal gain and the input and output impedances of filter circuits. Any non-linear models, such as diodes or transistors, are first linearized based on the DC bias point, and then the small signal DC analysis is performed.

Sensitivity Analysis

Produces the DC and AC sensitivities of an output variable with respect to all circuit variables, including model parameters.

The sensitivity function uses the direct approach [35] to support sensitivity calculations for the DC and AC analyses. The DC sensitivity is with respect to the DC operating point. SPICE calculates the difference in an output variable, either a node voltage or a branch current, by perturbing each parameter of each device independently. Since the method is a numerical approximation, the results may demonstrate second order effects in highly sensitive components, or may fail to show very low but nonzero sensitivity. Since each variable is perturbed by a small fraction of its value, zero-valued parameters are not analyzed. This analysis is useful when trying to find a worst case scenario of circuit operation. By finding the most sensitive components and moving their values accordingly, the circuit's performance can then be evaluated.

DC Analysis

Produces a series of DC operating points by sweeping one independent source or two sources in a nested loop.

The DC analysis is used in applications which are dependent upon static variables such as line regulation, load regulation, or the DC modulation gain of a power converter. The .DC function is a special subset of the DC analysis feature. It is used to perform a series of DC operating points by sweeping voltage and/or current sources and

performing a DC operating point at each step value of the source(s). At each step, the DC voltages, currents, and computed device/model parameters can be recorded. The DC statement defines the sources which will be swept, and their corresponding increments. One or two sources can be involved in the DC sweep. If two sources are involved, the first source will be swept over its range for each value of the second source. This option is useful for obtaining semiconductor device output characteristics or calculating load lines.

AC Analysis

Generates a frequency response/Bode plot of the circuit. Magnitude, phase, real, or imaginary data is produced.

The AC analysis is used to evaluate many performance characteristics which are covered in this book. It may be used to determine performance characteristics such as circuit stability, impedance, and filter attenuation.

The AC analysis in SPICE computes the small signal response of the circuit. Output variables are recorded as a function of frequency. Before the AC analysis is performed, SPICE first computes the DC operating point of the circuit. It then determines the linearized, small-signal models for all of the non-linear devices in the circuit, based on this operating point. The resultant linear circuit is then analyzed over the specified range of frequencies. It is very important to establish the proper DC circuit biasing in order for the AC analysis to produce useful data. For example, biasing an op-amp in its linear range will give different AC results than if the op-amp is saturated.

Although the AC analysis performs a sinusoidal steady state analysis, it should not be confused with a transient (time domain) analysis using a large signal SINE wave. The AC analysis is a small signal analysis in which all non-linearities are linearized. For instance, if the DC biasing of a transistor gain stage produces a gain of ten, then the gain will remain ten, regardless of the input value. If the input is 1, then the output will be 10. If the input is 100, then the output will be 1000. The gain is linearized. Under non-linear conditions, however, the gain of the transistor will roll off as the input is increased. The "VName 1 0 SIN..." stimulus is only used for non-linear time domain analyses, and should not be confused with the "Vname 1 0 AC 1" frequency response stimulus.

Frequency mixing note: The AC analysis is a single frequency analysis. Only one frequency is analyzed at a time. Therefore, circuits which perform signal mixing will not benefit from the AC analysis. In order to see frequency mixing, you will have to run a transient analysis and

convert the output waveforms into the frequency domain using a Fourier transform.

Transient Analysis

Runs a non-linear time domain simulation.

The transient analysis computes the circuit response as a function of time over any time interval. Output data, including node voltages and voltage source currents, can be recorded. During a transient analysis, numerous independent sources may have active time varying stimulus signals.

It is often necessary to start a SMPS simulation with a pre-defined set of operating conditions. The UIC (use initial conditions) keyword in the .TRAN statement causes SPICE to skip the initial transient solution (operating point) which is normally performed prior to the transient analysis. If this keyword is included, the values which are specified via "IC =" specifications on the various elements and .IC statements are used as the sole source for initial conditions. The transient analysis will begin with these values.

Fourier Analysis

Fourier analysis provides a simple means for evaluating the harmonic content of a time domain waveform. This analysis may be used to determine performance characteristics such as the conducted emissions performance of a switching power supply, or the harmonic content of a sine wave output converter. A Fourier analysis can be performed by SPICE, but is usually performed using a separate data post processing program which operates on the .PRINT transient simulation output data.

Temperature Analysis

SPICE allows the temperature of the circuit, or a particular element, to be varied.

SPICE simulates circuits using a global temperature of 27°C. This can be changed using the .OPTION TEMP= statement. In addition, SPICE 3 allows you to set the temperature for an individual device. This feature permits the simulation of a temperature gradient, as well as a "hot" device. Individual device temperatures are set directly on the device call line or in the .Model statement.

While the Monte Carlo, worst case, and optimization analyses are not inherently part of SPICE, most commercial vendors have added

them to the list of simulation capabilities. They are an invaluable part of SMPS investigation and design.

Monte Carlo and Worst Case Analysis

The Monte Carlo tolerance analysis is an ideal application for circuit simulation. The effects of component tolerance variations are difficult to assess by any other means. Imagine sitting in an engineering lab and sorting resistors, capacitors and other components in an attempt to find the worst case tolerance extremes to place in your circuit.

This investigation is usually performed either as a worst case analysis, or as a Monte Carlo Analysis. These analyses seem to be used interchangeably, although they are quite different.

A Worst Case analysis determines the worst case performance, but does not determine the statistical weighting of performance. As a general rule, the worst case analysis is preferred if the worst case values can be easily determined. In many cases, however, it is difficult to know which components must be varied, and in which direction, in order to generate the worst case result.

A Monte Carlo analysis provides the statistical weighting, but does not provide the worst case result. Monte Carlo analysis is generally used to calculate the mean and standard deviation of a particular performance characteristic. This analysis takes significantly longer to run than the worst case analysis, since it requires many simulations.

Optimizer Analysis

The Optimizer analysis is a powerful IsSpice feature that allows a series of simulations and measurements to be automatically performed over a range of component values. Either one or two circuit variables may be swept through a specified range of values. This feature is excellent for determining the damping components of an EMI filter.

**This Page is Inserted by IFW Indexing and Scanning
Operations and is not part of the Official Record**

BEST AVAILABLE IMAGES

Defective images within this document are accurate representations of the original documents submitted by the applicant.

Defects in the images include but are not limited to the items checked:

- ☐ **BLACK BORDERS**
- ☐ **IMAGE CUT OFF AT TOP, BOTTOM OR SIDES**
- ☒ **FADED TEXT OR DRAWING**
- ☐ **BLURRED OR ILLEGIBLE TEXT OR DRAWING**
- ☐ **SKEWED/SLANTED IMAGES**
- ☐ **COLOR OR BLACK AND WHITE PHOTOGRAPHS**
- ☐ **GRAY SCALE DOCUMENTS**
- ☐ **LINES OR MARKS ON ORIGINAL DOCUMENT**
- ☐ **REFERENCE(S) OR EXHIBIT(S) SUBMITTED ARE POOR QUALITY**
- ☐ **OTHER:** _____

IMAGES ARE BEST AVAILABLE COPY.

As rescanning these documents will not correct the image problems checked, please do not report these problems to the IFW Image Problem Mailbox.